NASA Contractor Report 172496

Numerical Approach for the Aerodynamic Analysis of Airfoils With Laminar Separation

Prepared by: David W. Halt and Dean R. Bristow

McDonnell Douglas Corporation
McDonnell Aircraft Company
P.O. Box 56
St. Louis, Missouri 63166

Contact NAS1-17466
February 1985

NASA

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665

NASA Contractor Report 172496

Numerical Approach for the Aerodynamic Analysis of Airfoils With Laminar Separation

Prepared by: David W. Halt and Dean R. Bristow

McDonnell Douglas Corporation
McDonnell Aircraft Company
P.O. Box 56
St. Louis, Missouri 63166

Contact NAS1-17466
February 1985

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665

N85-18001#

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF ILLUSTRATIONS (Concluded)

SUMMARY

The objective of this research is to formulate and evaluate a new numerical method for simultaneously and efficiently coupling an external subsonic potential flow and an interior viscous flow such that the two flows match at an interfacing boundary. Both a panel method and a simple point compressible vortex model are used for the outer potential field. The interior flow solvers which were used are the Navier-Stokes and Euler codes of T. J. Coakley (NASA Ames) and the Euler code of A. Verhoff (MCAIR). In order to test compatibility, the panel method is coupled to the less expensive Euler codes since the coupling procedure is identical to the Navier-Stokes code.

The results of this study show significant efficiency improvements can be obtained over the uncoupled approach. Results also indicate the outer potential flow is best represented by the simple point compressible vortex model. The panel method couples smoothly to Coakley's implicit code but is numerically incompatible as coupled with MCAIR's explicit Euler code.

An improved Navier-Stokes code is under initial development which extends MCAIR's Euler code to include the necessary viscous terms. Results are shown for an infinite length channel with one wavy periodic wall with and without laminar separation. Further development is recommended for airfoil applications with the point compressible vortex used for the outer potential flow region.

SECTION I
INTRODUCTION

The primary objective of this effort is to develop an efficient coupled viscous/inviscid flow solver capable of accurately analyzing airfoil flows with laminar separation. A numerical method is formulated and evaluated which couples an external subsonic potential flow and an interior viscous flow such that the two flows match at an interfacing boundary located outside the region where viscous effects are important (see Figure 1). An existing Navier-Stokes Equation solver is used to compute the flow field for the inner viscous region including flow values on a paneled boundary. The outer region is governed by the LaPlace equation where all flow quantities can be determined from the velocities on the surface of the paneled boundary. The matching problem is to solve for all flow quantities inside and on the panel boundary using the Navier-Stokes flow solver with outer boundary values determined by the coupled panel method.

Objective

- Predict Effect of Laminar Separation on Subcritical Performance
- Efficient Time Dependent Solutions at Realistic Reynolds Number

Approach

1) Navier-Stokes Equation Solver
2) Paneled Interface Boundary
3) External Flow From Panel Singularities

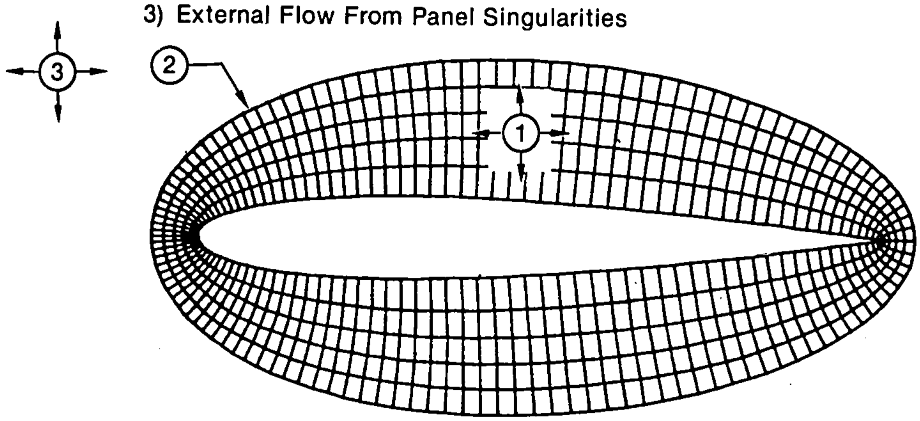


Figure 1. Numerical Approach for Airfoils With Laminar Separation

This contract is part of a larger joint effort as shown in Figure 2 to analyze airfoils undergoing laminar separation. McDonnell Aircraft Company (MCAIR) had developed under this contract a coupled potential/Navier-Stokes method and evaluated it for accuracy and efficiency. R. J. Margason of NASA LaRC served as technical monitor for this effort as well as for the complementary research of Drs. K. N. Ghia, U. Ghia and G. A. Osswald of the University of Cincinnati (Reference 1).

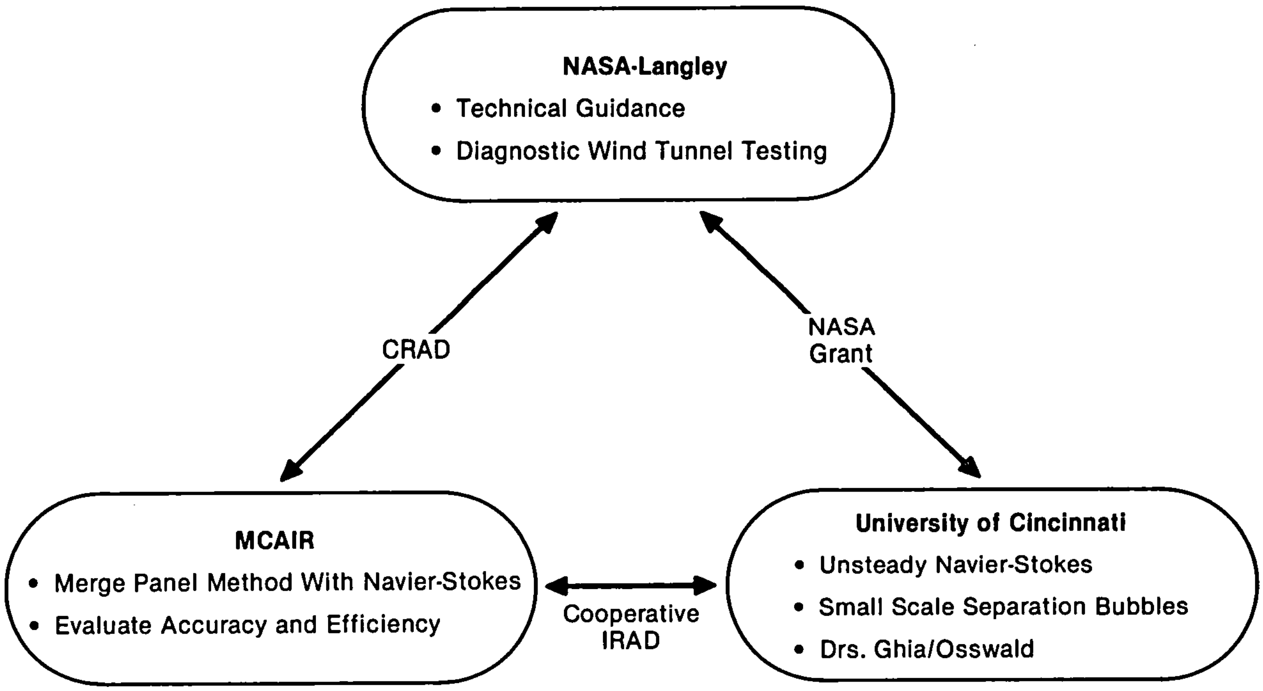


Figure 2. NASA-Industry-University Cooperative Research Effort Analytical Method for Laminar Separation

The work of B. R. Gilmer and D. R. Bristow, MCAIR (Reference 2) serves as motivation to the MCAIR effort. A fully coupled integral boundary layer/panel method was developed to analyze airfoils at high angle of attack undergoing massive turbulent separation. Three representative solution sets are shown in Figure 3 for differing Reynolds Number. Results agree very well with test data (Reference 3). The advantage of using a Navier-Stokes equation solver in place of a boundary layer method is to increase applicability to more general flows. Airfoils undergoing laminar separation with possible reattachment are of particular interest to this study. This report describes our effort to develop such a coupled approach.

Section II describes the formulation of the necessary panel relations between outer boundary velocities and panel velocities. Several approximations are described based on potential flow theory. The accuracy of the panel method is addressed in relation to applicable locations of the panel boundary.

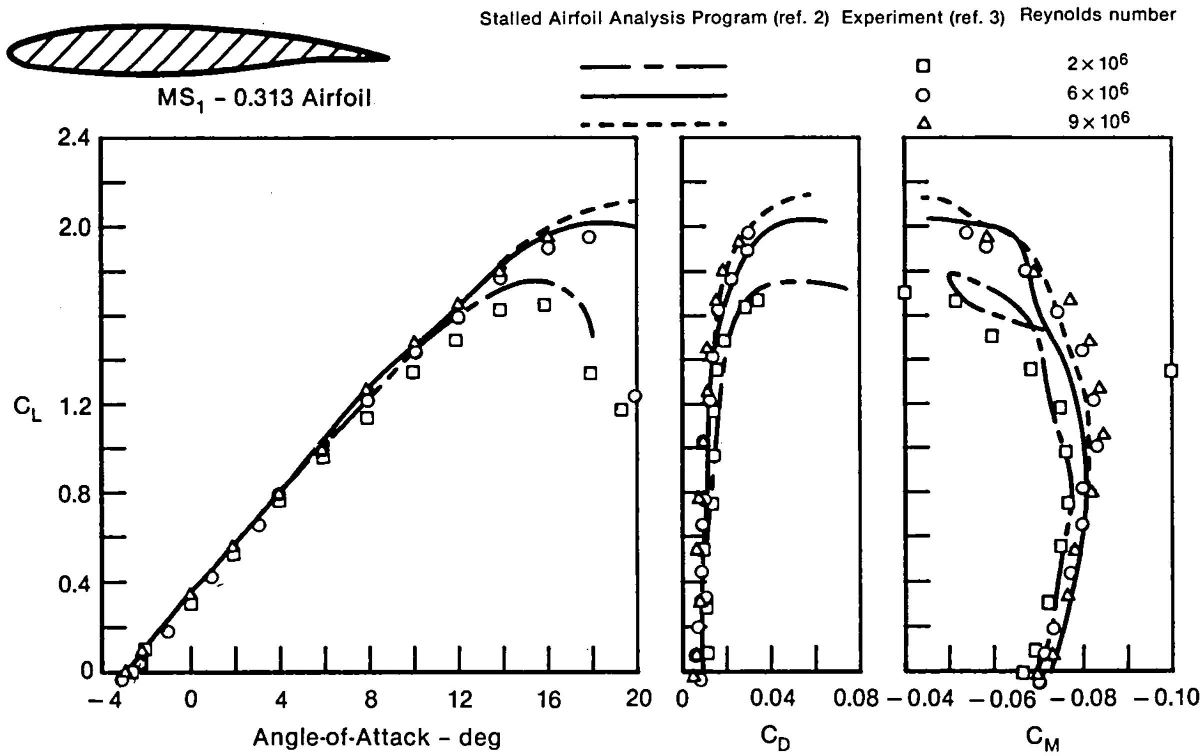


Figure 3. Fully Coupled Boundary Layer/Panel Method Capability

The coupling procedure is discussed in Section III. The interior flow solvers used in this study are the Navier-Stokes and Euler codes of T. J. Coakley, NASA Ames (References 4-6) and the Euler code of A. Verhoff, MCAIR (Reference 7). The panel method is coupled only to the Euler codes in order to reduce developmental cost. Coupled results are shown using Coakley's Euler code which indicate a significant increase in accuracy over the uncoupled code. A numerical incompatibility results between the panel method and the explicit Euler code of MCAIR as presently coupled. An alternative to the present coupling procedure is discussed.

A simplified outer potential field concept is presented in Section IV. A simple point compressible vortex is used to model the effect of airfoil circulation on the outer boundary as suggested by J. L. Thomas, NASA LaRC (Reference 8). The strength of the vortex is based on the airfoil pressure integrated lift coefficient, thus being applicable to viscous as well as inviscid flows. Results are shown which indicate the current coupling procedure with the panel method shows no benefit over the simple point vortex model of an outer potential field.

The approach that appears most promising for the analysis of airfoils with laminar separation is to add viscous terms to the MCAIR explicit Euler method while using the point compressible vortex for the outer flow region. Preliminary development of a laminar Navier-Stokes code is presented in Section V. The formulation is based on Riemann Variables in local streamwise coordinates. The advantages of this approach are discussed and preliminary results are given for infinite length wavy wall channel flows with and without laminar separation. Further development is recommended for airfoil applications.

All calculations for this study were performed on the NASA LaRC computing complex. Most flow calculations were performed on the CYBER 203 vector machine.

SECTION II
THE PANEL METHOD

In this section, the fundamentals of the panel method are explained, the accuracy of the panel method in reproducing outer velocities computed by the Navier-Stokes flow solver is estimated, and the extent to which the outer potential region may approach the airfoil surface is determined approximately. The matching process between the panel method and the interior flow solver will be described in the next section.

The panels are interconnected along a chosen C-grid line and closed along a vertical grid line downstream as shown in Figure 4. The objective is to establish panel singularity strengths that nearly reproduce the velocity vectors on the panels. It is not generally possible to establish a potential field that will reproduce these velocities since the panel boundary is influenced by viscous and rotational effects. Either set of tangential or normal velocity components alone will define unique potential fields.

The outer potential field may be defined by one of the three choices shown in Figure 4. The panel boundary value problem is then easily set up and solved using Morino type boundary conditions (Reference 9). Given N panels, it is sufficient to solve N-1 control point equations along with an auxiliary relationship in order to define the potential outer region.

$$\phi_{i+1} - \phi_i = o = \sum_j \left[(A_{\sigma_{i+1,j}} - A_{\sigma_{i,j}})\, \sigma_j + (A_{\gamma_{i+1,j}} - A_{\gamma_{i,j}})\, \gamma_j \right]$$

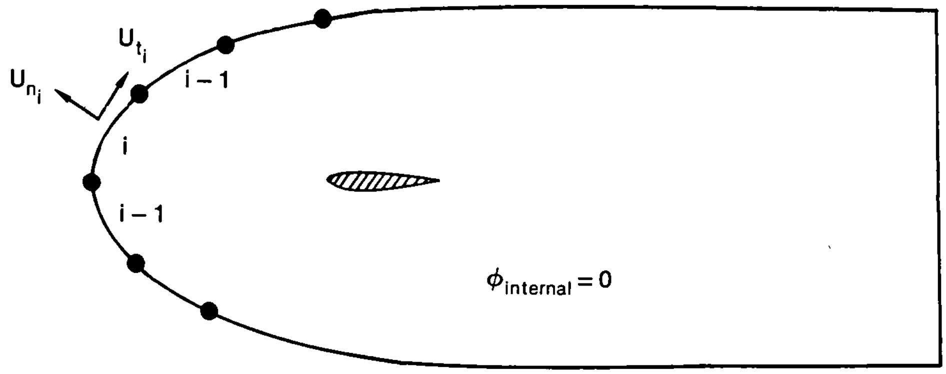
$$i = 1,2,\ldots N-1$$

The Nth equation satisfies zero total mass flux through the panel boundary when the outer potential is defined by tangential panel velocity components.

$$0 = \sum_j \Delta S_j \sigma_j$$

Panel length is represented by ΔS. When the potential is defined by normal panel velocities, the Nth equation controls the circulation on the panel boundary.

$$\Gamma = \sum_j \Delta S_j \gamma_j = \sum_j \Delta S_j U_{t_j}$$



$\{\phi_{int}\} = \{0\} = [A_\sigma]\{\sigma\} + [A_\gamma]\{\gamma\}$

σ_i = Local Source Strength

γ_i = Local Vortex Strength.

Choice of Outer Potential Flowfields:

1. $\{\gamma\} = \{U_t\}$; $\{\sigma\} = -[A_\sigma]^{-1} \cdot [A_\gamma]\{\gamma\}$ (Defined by Tangential Velocities)
2. $\{\sigma\} = \{U_n\}$; $\{\gamma\} = -[A_\gamma]^{-1} \cdot [A_\sigma]\{\sigma\}$ (Defined by Normal Velocities)
3. Some Combination of 1. and 2.

Figure 4. Definition of the Outer Potential Flow Region by the Panel Method

The influence coefficients are constructed by the methods given in Reference 10 and are of the form:

$$U^+_{t_i} = \sum_j \left[D_{\sigma_{ij}} \ \sigma_j + D_{\gamma_{ij}} \ \gamma_j \right]$$

$$U^+_{n_i} = \sum_j \left[E_{\sigma_{ij}} \ \sigma_j + E_{\gamma_{ij}} \ \gamma_j \right]$$

where the "+" superscript represents potential region values exterior to the panel boundary.

Substitution of the appropriate relations shown in Figure 4 yields the combined influence coefficients (B_{ij}, C_{ij}). Given the first choice of potential fields we have:

$$U^+_{t_i} = \sum_j \left[-\ (D_\sigma \ A_\sigma^{-1} A_\gamma)_{ij} + D_{\gamma_{ij}} \right] \gamma_j = \sum_j B_{ij} \gamma_j$$

$$U^+_{i} = \sum_j \left[-\ (E_\sigma \ A_\sigma^{-1} A_\gamma)_{ij} + E_{\gamma_{ij}} \right] \gamma_j = \sum_j C_{ij} \gamma_j$$

Any of the three choices of outer potential will exhibit a velocity mismatch between the two regions since the inner region is generally nonpotential. The potential region can only be expected to match one velocity component along the panel boundary. This problem can be eliminated if we can accept a non-potential field generated as follows. The outer tangential velocities are computed using tangential panel velocities to define the singularities, whereas the outer normal velocities are computed using normal panel velocities to define their own singularities. The resulting outer region is generally non-potential since the two potential fields are selectively combined using appropriate velocity components from each potential field. The advantage to this approach is a smooth match between velocities along the panel boundary.

The influence coefficients based on panel tangential velocities are computed using linear vortex and linear source panel distributions, whereas those based on normal velocities use linear vortex and constant source distributions. The influence coefficients need to be computed only once before the Navier Stokes iteration process begins. After each iteration the outer velocities can be updated using the above relations. Region 1 is matched with Region 3 (refer to Figure 1) when the panel velocities determine the outer velocities which in turn satisfy the Navier-Stokes boundary value problem.

A complete coupling of panel boundary conditions with the Navier-Stokes computer code was not attempted at this time. Instead it is instructive to consider a comparison between the velocities on a desired grid line from a computed Navier-Stokes solution with those same velocities recomputed by the previous panel relations along a chosen paneled boundary. This allows us to observe the compatibility of numerical approximations between the Navier-Stokes code and the panel method.

A Navier-Stokes solution is computed using Coakley's code (Reference 6) for a NACA 0012 airfoil at two degrees angle of attack and freestream Mach Number of 0.3 given a 160 by 50 C-grid. The velocities are recorded along every grid line surrounding the airfoil. Approximately every third point in each grid line is compared with recomputed values using the panel relations based on an inner paneled grid line. For each case the inner paneled grid line is located three cells inside the grid line of interest. In Figure 5, the horizontal and vertical velocities (U_x, U_y) are compared for the 40th grid line from the airfoil. This grid line is located approximately one chord length from the airfoil surface and extends ten chord lengths downstream from the trailing edge. The plotted velocities are

perturbations from freestream in order to magnify the deviation between Navier-Stokes and potential velocities. The comparison at this station between outer velocities is good with two notable exceptions. First, a significant discrepancy occurs in the U_x perturbations along the wake of the airfoil. Secondly, a residual vertical velocity error is apparent along the outer boundary. The overall comparison indicates the panel approximation can serve as a good boundary value approximation for the interior flow solver.

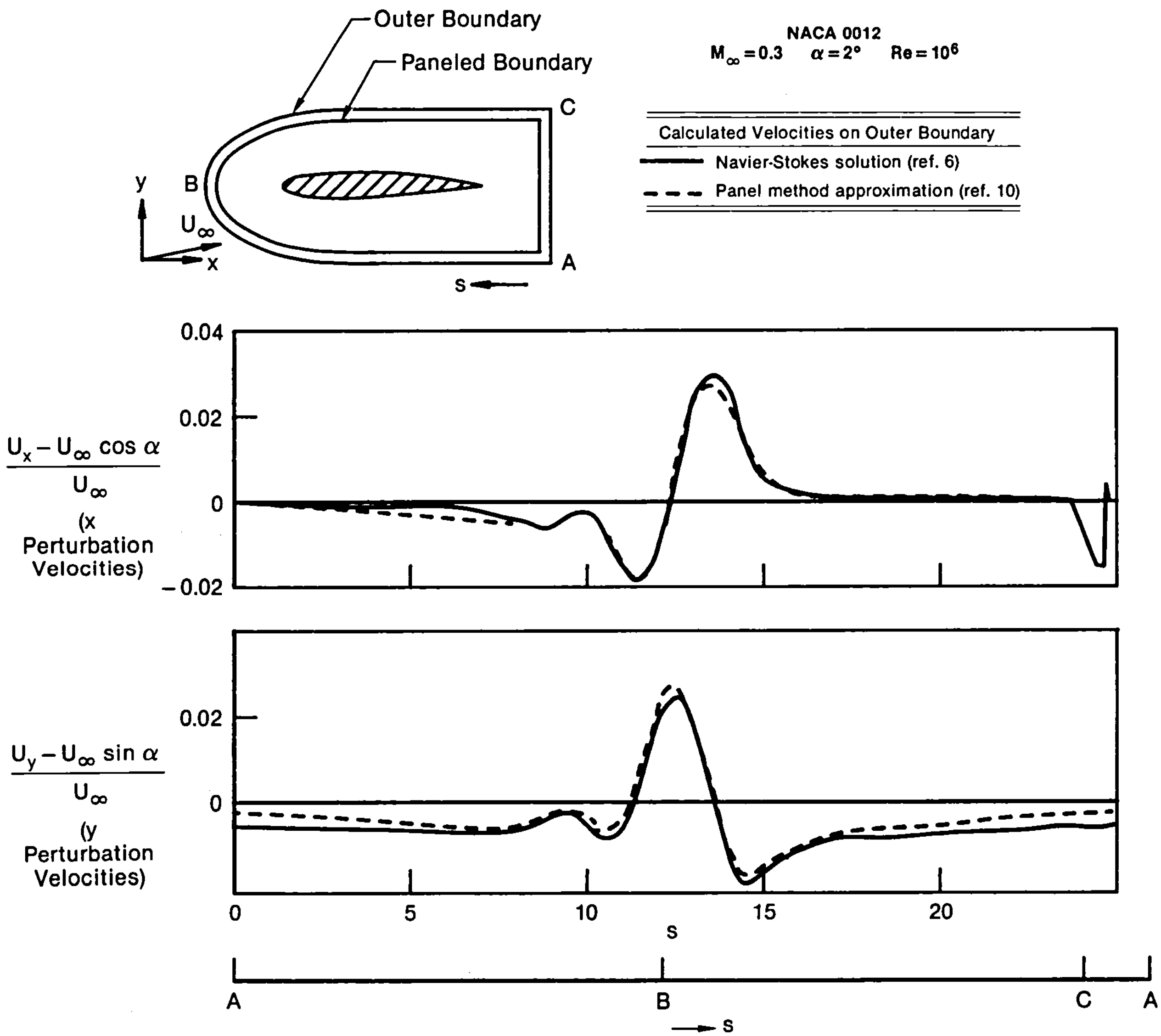


Figure 5. Accuracy of the Panel Method Approximation

A comparison of the panel approximation for the ten recorded grid lines is shown in Figure 6. An RMS error is computed relating Navier-Stokes velocities to velocities of the panel method for each grid line and plotted versus the height between grid lines, h. The error is computed as,

$$\text{RMS ERROR} = \sum_i \frac{\Delta S_i}{S_{TOT}} \left[(U^+_{t_{pan}} - U^+_t)^2 + (U^+_{n_{pan}} - U^+_n)^2 \right]^{\frac{1}{2}}_i$$

where S refers to arc length along a grid line. For example, the RMS error for the 40th grid line (refered to in Figure 5) is given at the height-to-chord ration of two. As we approach the airfoil, the error grows rapidly due to the significant increase in diffused vorticity and relative magnitude of the perturbation velocities. The relation between h and the total number of grid points in Region 1 is a measure of efficiency since the amount of computational work is nearly proportional to the number of grid points. Work done by the panel method is negligible. The figure indicates a reduction of nearly 25 percent of the workload may be achieved with a minimal loss of accuracy by use of the coupled approach. This may be accomplished by eliminating about two thousand grid points in the outermost region.

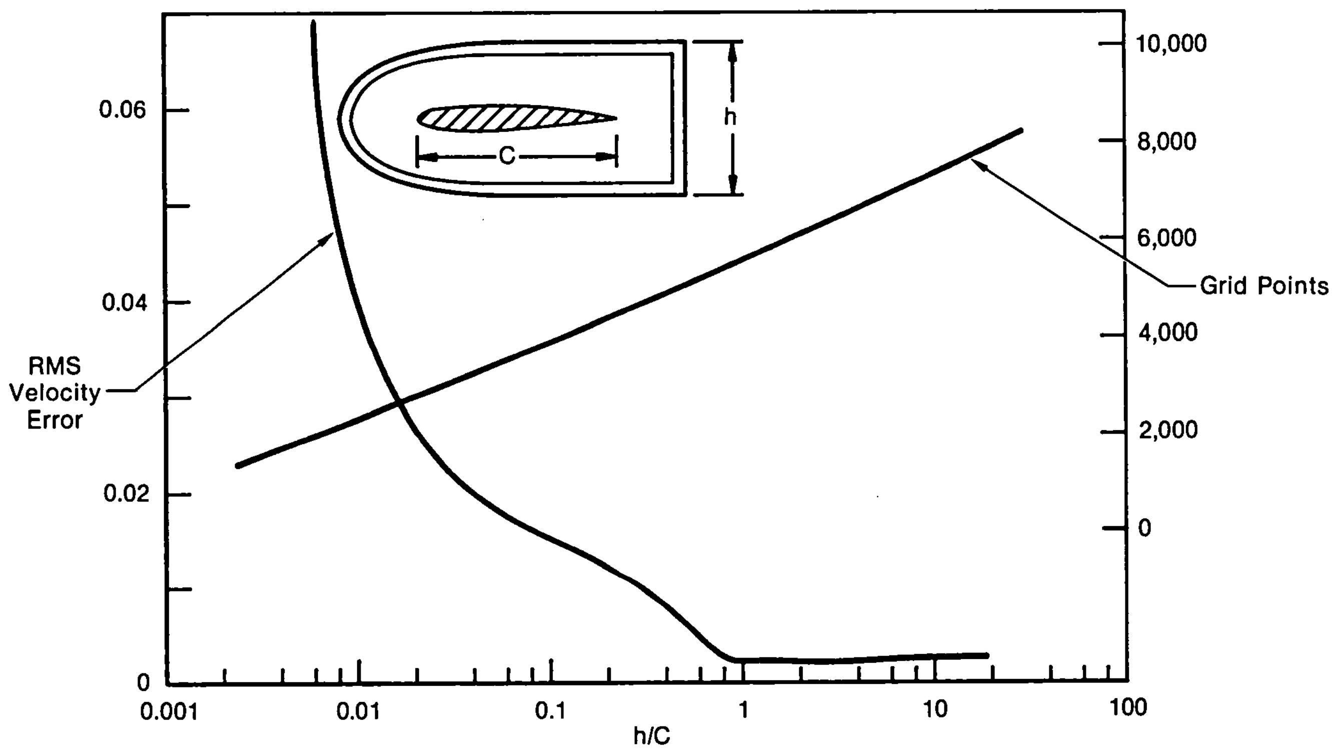


Figure 6. A Study on the Effect of Accuracy and Efficiency by Varying the Extent of the Outer Potential Flow Region

SECTION III
COUPLING THE PANEL METHOD TO AN INTERIOR FLOW SOLVER

A detailed description of the coupled approach between the panel method and two different interior flow solvers is given in this section. The selected interior flow solvers are both Euler codes in an effort to reduce development cost. The actual coupling process is identical between the panel method and either the Euler or Navier-Stokes version of the same code since the outer field is considered inviscid. Results are given and the various anomalies of the approach are discussed.

The coupled iteration cycle is shown in Figure 7. The entire process starts with preliminary setup of grid geometry, initial flowfield and calculation of panel influence coefficients described in the previous section. The iteration cycle then begins with an unmodified interior flow solver iteration (e.g. Navier-Stokes) using freestream boundary conditions on the outer grid lines. Panel singularities are then computed along a preselected panel boundary. This step is trivial since only one set of singularities corresponding to either tangential or normal perturbation panel velocities are needed to define the outer potential field. The velocities on the outer boundary of the discretized flow solver are then computed using the precalculated influence coefficients. Constant enthalpy and entropy conditions are used to determine the remaining two boundary values on the outer grid line. The iteration cycle repeats with continually modified outer boundary values until convergence.

The Euler problem would be overspecified if all four boundary values were specified as Dirichlet conditions. Only those boundary values that correspond to incoming characteristic waves are specified as shown in Figure 8 for Coakley's code. Namely, three flow values are specified at an inflow boundary and only pressure is specified at an outflow boundary. This is consistent with the one dimensional approximation of the wave equations as described by Coakley (Reference 4). The remaining boundary values which correspond to outgoing characteristic waves must satisfy the compatibility relations.

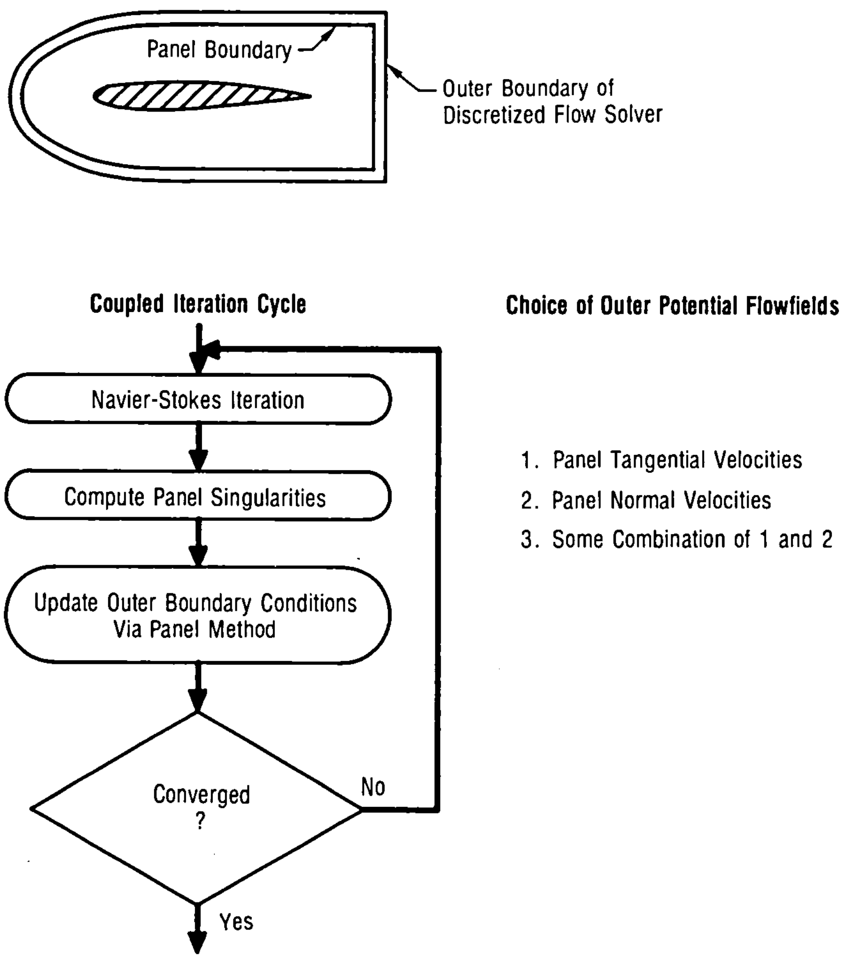


Figure 7. The Coupled Navier-Stokes/Panel Method Approach

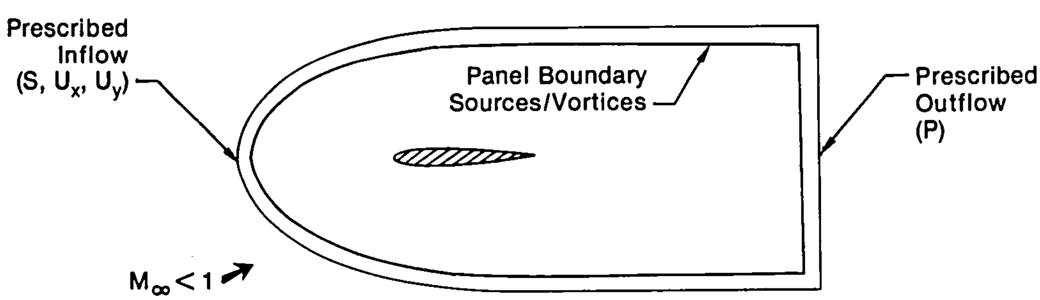


Figure 8. Coupling the Panel Method With Coakley's Implicit Euler/Navier-Stokes Code

The coupled method was applied to the NACA 0012 airfoil case at Mach 0.3 and 10 degrees angle of attack. Several calculations of lift coefficients are compared in Figure 9 varying the location of the outer boundary. The outer boundary radius is reduced simply by truncating the outer grid lines from the original 120 by 30 C-grid. This enforces uniformity in the grids near the airfoil among the various cases. The outer velocities were calculated from both sets of panel velocity components as discussed in the previous section. The same cases were recomputed with the original Coakley code which uses freestream prescribed boundary values.

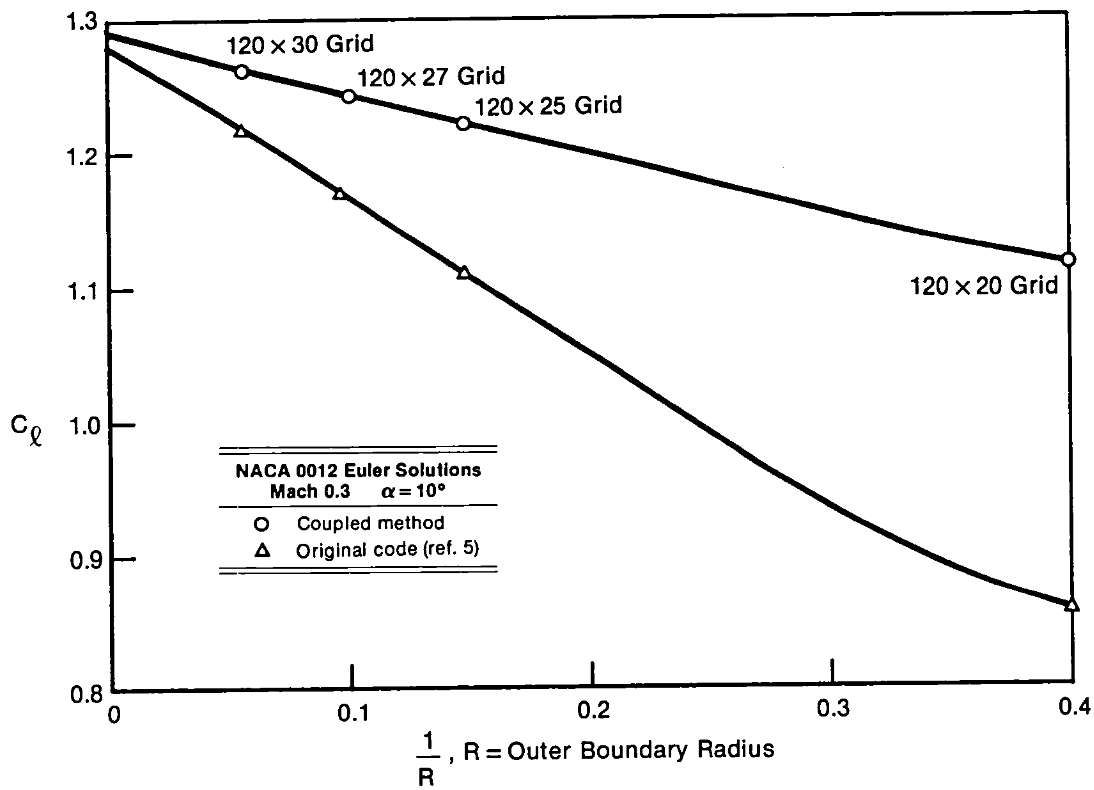


Figure 9. Comparison of Lift Coefficient for Various Outer Boundary Locations

Results are shown for four grid sizes ranging from a far field radius of 2.5 chord lengths to 20. Each C-grid extends 10 chord lengths downstream from the airfoil. Both methods approach nearly the same lift coefficient as the outer boundary radius approaches infinity. The limiting lift coefficients should not be expected to be exactly the same since the backplane is located 10 chord lenths downstream from the airfoil in all cases. The coupled approach gives a more accurate lift coefficient for each case. The difference is magnified as the outer boundary radius is reduced. A major concern at this point is the variance in lift coefficient for the coupled method.

One of the more apparent reasons why the lift varies for the coupled approach is that circulation is not conserved in the Euler code. Figure 10 compares the circulations expressed as local lift coefficients for each C-grid line connected by the vertical grid line far downstream. The circulation should not vary for subcritical inviscid flows, but there exists a numerical error for this case. Since the circulation varies with grid line, it is apparent the choice in location of the panel boundary affects the circulation imposed on the outer boundary. The calculated lift coefficients for varying panel location are compared in Figure 11, indicating a considerable discrepancy.

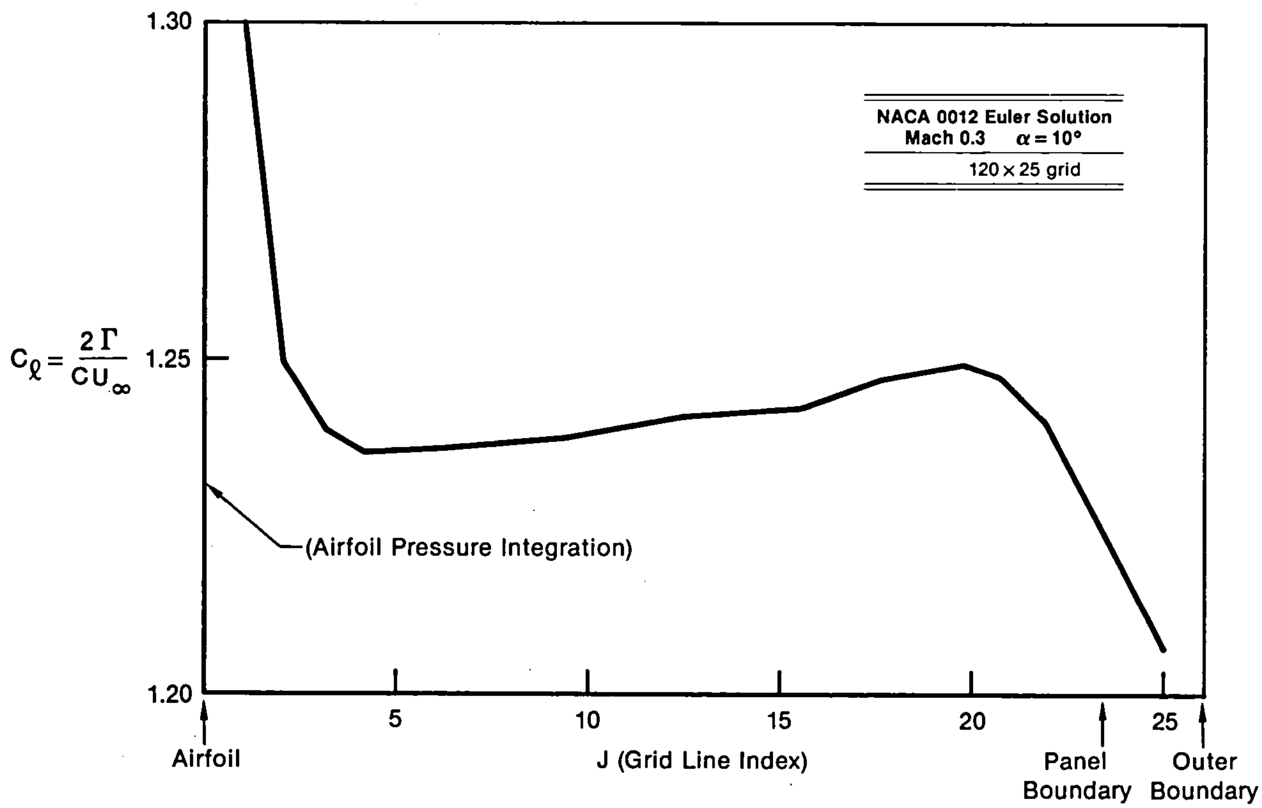


Figure 10. Circulation Variation on C-Grid Lines

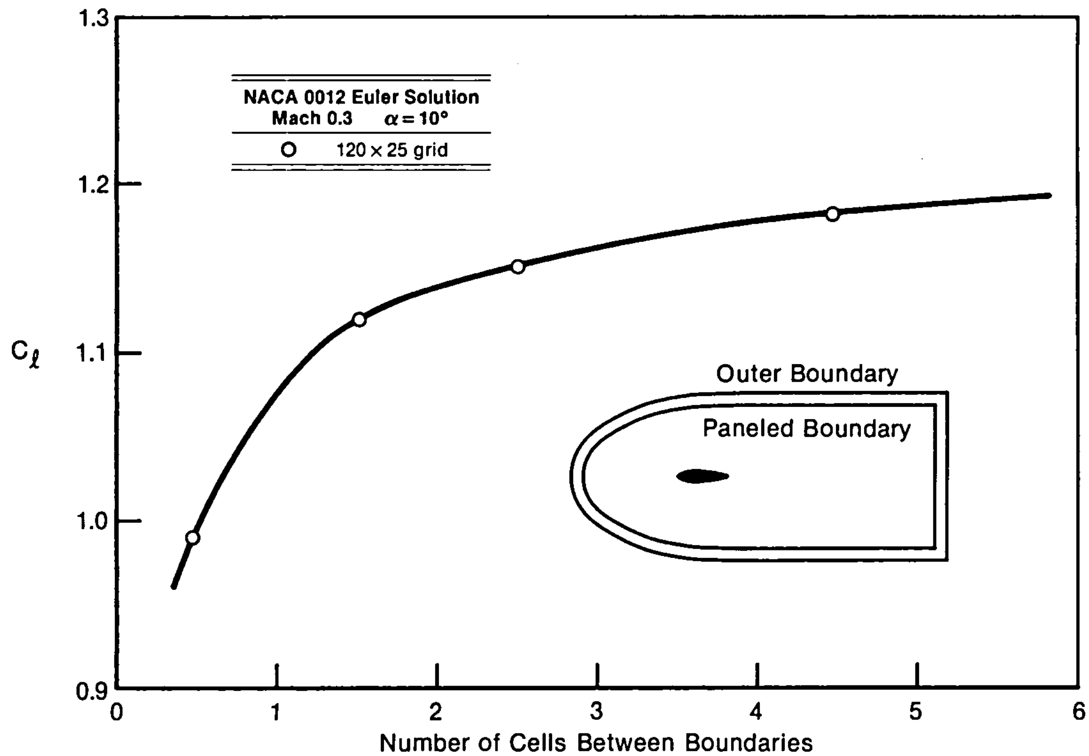


Figure 11. The Effect of Panel Location on a Fixed Outer Boundary

Since the outer potential region starts at the panel boundary, it is desirable to locate the panels near to the outer boundary. However, when the panels are one half cell width inside the outer boundary, a conflict between the panel relations and the compatibility relations is introduced. For example, on the downstream boundary the compatibility relation prescribes a zeroeth order extrapolation to determine tangential velocity, whereas the panel relations may call for a significant change. This compatibility relation apparently becomes less of a conflict as the panels are moved nearer to the airfoil.

The above calculations were made using panel relations between tangential velocities on the two boundaries and similar panel relations between normal velocities. The resulting outer flow is generally non-potential since a different potential field was used to compute each velocity component. Panel relationships based only on tangential panel velocities have been used, but results are similar to those shown earlier.

The panel method is similarly coupled to MCAIR's explicit Euler code (Reference 7) for a second comparison of the generalized coupled approach. The boundary conditions are easily implemented in this code because the characteristics approach with local streamwise coordinates clearly leads to a straightforward outer boundary condition. The prescribed Dirichlet boundary values are shown in Figure 12 for this extended Riemann flow solver. The coupling procedure between panel method and Euler method is similar to that with the previous Euler code.

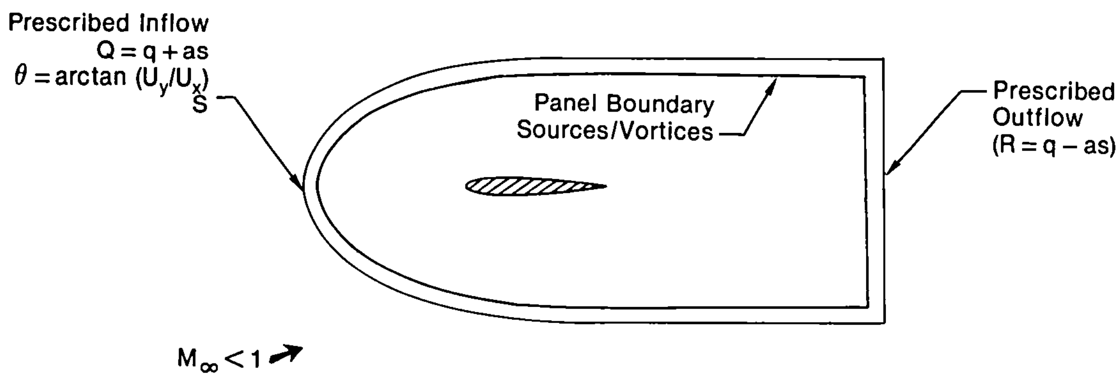


Figure 12. Coupling the Panel Method With Explicit Euler Code

The coupling of the panel method to the explicit code, however, results in a neutral instability which apparently is controlled only by reducing the frequency of updated outer boundary values. Figure 13 shows a NACA 0012 solution history in which the panel method was used every iteration to update the appropriate boundary values. This solution is clearly unacceptable since the circulation does not approach a steady state. The same features are present when the outer boundary values are updated every two hundred iterations, but the magnitude of the oscillations has decreased. Convergence to steady state is achieved only after five hundred or more iterations are performed before each update of the boundary values. The rate of convergence, however, is slowed after each implementation of updated boundary values. A comparison of the lift coefficients between the coupled method and the original code were not obtained because of the unwieldy convergence of the coupled method. Other attempts to alleviate the problem were unsuccessful including under-relaxation, panel boundary movement, and different methods of generating the outer potential field. The reason for this instability may be the incompatibility between the panel method where information travels instantaneously to every other point in the potential field and the explicit Euler code where information can only propagate one cell length per iteration.

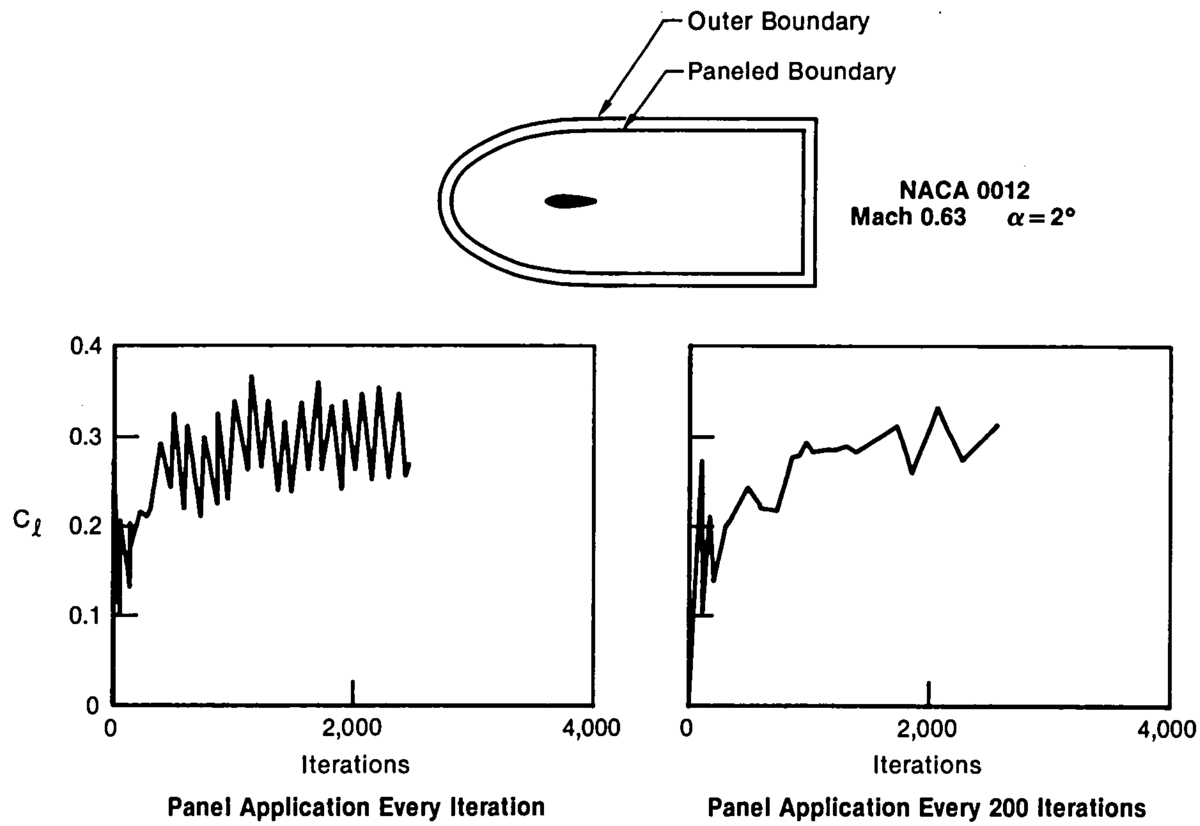


Figure 13. Panel Method Coupled With Explicit Euler Code
Demonstration of Convergence Instability

One of the difficulties with the general coupling procedure used in this study is associated with the multiple outer boundary concept. Another approach would be to form one outer boundary by locating the panel boundary along the outer grid boundary. The problem would be altered to determine the appropriate prescribed values which best approximate an outer potential field in a least squares sense. The outflow prescribed value, R, shown in Figure 12 can be determined simply from the local values of Q and S using a constant freestream total enthalpy. At inflow, Q may be calculated similarly from R and S. Entropy prescribed at inflow would simply be the freestream value which leaves the set of incoming flow angles, θ, as the only unknowns. Unfortunately, this approach requires the solution of a large full matrix problem every iteration unless this matrix is grossly approximated by its diagonal terms only. This particular approach is not recommended for efficiency reasons; however, an investigation of this approach could lead to a better understanding of a properly coupled method.

SECTION IV
A SIMPLIFIED OUTER POTENTIAL REGION

A viable alternative to coupling a panel method with a Navier-Stokes solver is to simplify the outer potential flow method. Since the lift coefficient varies considerably for the coupled method shown in Figure 9, it is desirable to maintain a large outer boundary radius to obtain the most accurate results. At large radius we can simplify the outer potential field by modeling the effect of the airfoil on the outer boundary with a simple point compressible vortex (Reference 8) located at quarter chord position. This method is coupled to Coakley's airfoil code as shown in Figure 14. The strength of the vortex is calculated from freestream velocity and pressure integrated lift coefficient. The appropriate outer boundary values are calculated from superposition of the point vortex in the freestream flow with auxiliary relations of freestream total enthalpy and entropy.

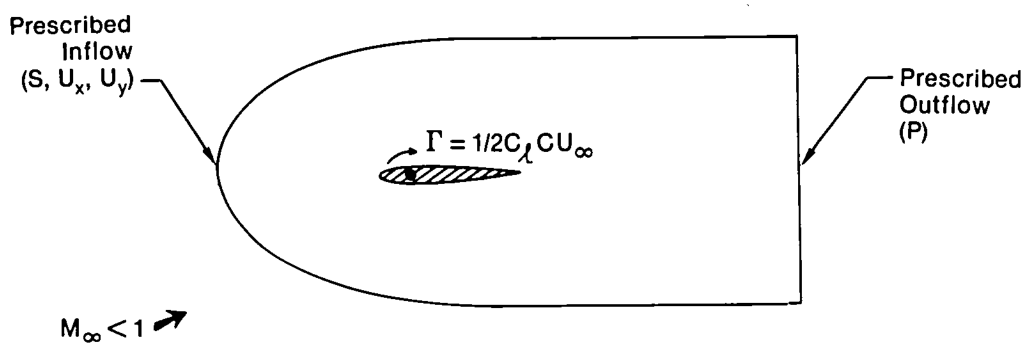


Figure 14. Simplified Outer Potential Field
Point Compressible Vortex

The NACA 0012 case at Mach 0.3 and ten degrees angle of attack is revisited using the coupled point vortex approach. The same four outer boundary radius solutions are calculated and compared with earlier results in Figure 15. The point vortex solutions are slightly favorable to the coupled panel method and significantly better than the original code results. The point compressible vortex is simple to incorporate in the Euler/Navier-Stokes code and requires little computational overhead. The coupled panel method requires storage of two influence coefficient matrices (N by N) and uses approximately two percent more CPU time per iteration for a given mesh size as compared to the original code. The coupled point vortex approach has produced better results than the coupled panel methods discussed previously. It is simple, reliable, and more accurate for near field outer boundaries.

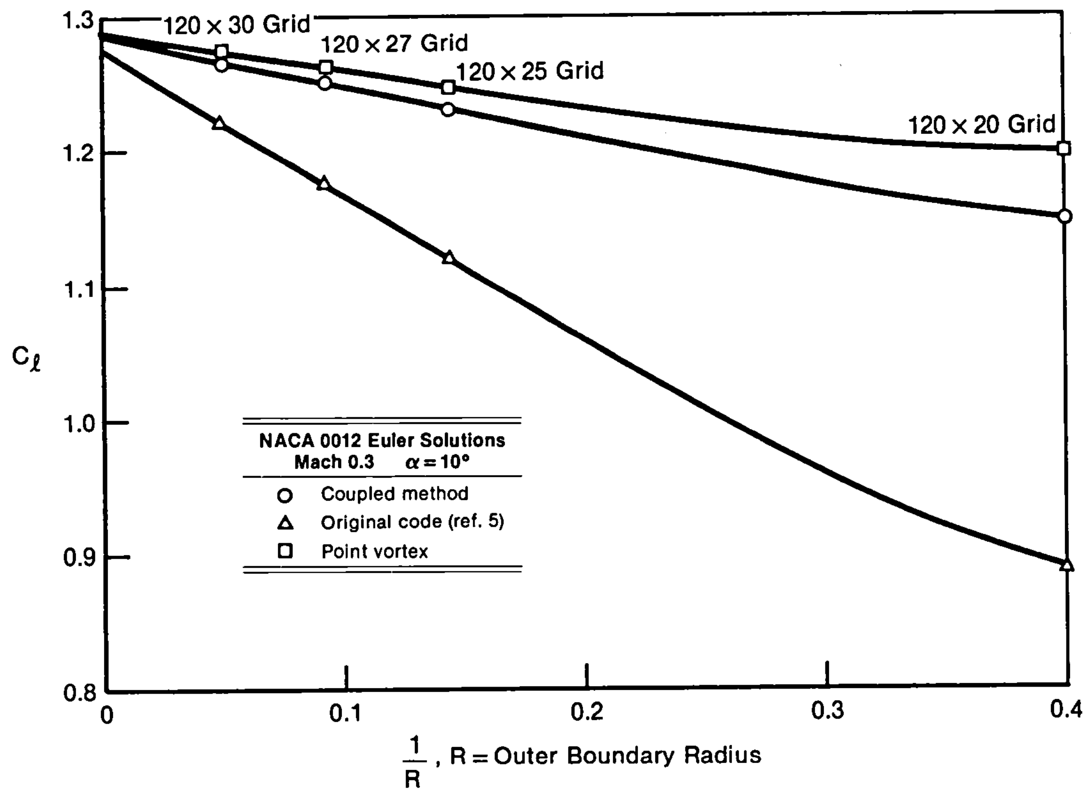


Figure 15. Comparison of Outer Potential Flow Region Methods

The point compressible vortex approach works equally well with Coakley's Navier-Stokes solver. Several airfoil cases have successfully been computed with this approach. A transonic RAE 2822 airfoil case is shown in Figure 16 for comparison between the original boundary conditions and modified values using the present approach. The lift as expected is significantly higher indicating the same probable trend as shown earlier. The point compressible vortex approach appears most favorable for analyzing airfoil flows with laminar separation and should work well with any basic flow solver.

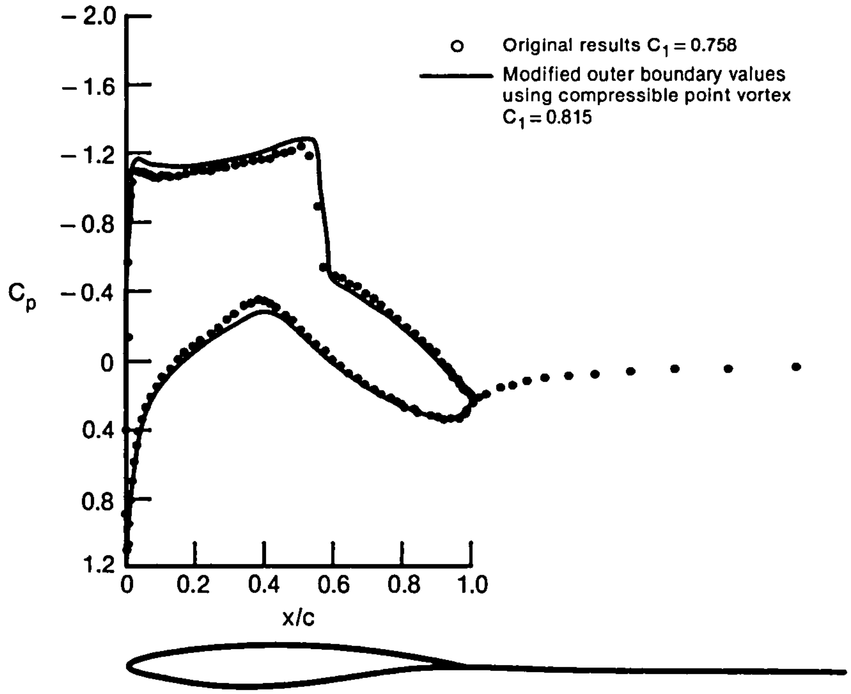


Figure 16. RAE 2822 Airfoil Navier-Stokes Solution
Q-Omega Turbulence Model 160 × 50 Grid
Mach 0.73 $\alpha = 2.78°$

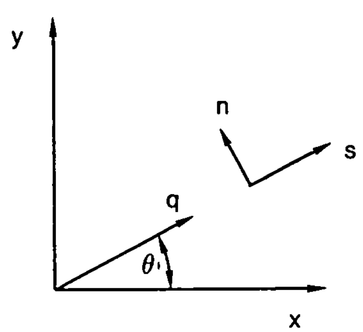
SECTION V
IMPROVED NAVIER-STOKES DEVELOPMENT

A laminar Navier-Stokes equation solver is under development in an effort to improve the accuracy and reliability in comparison to available codes. An approach that appears promising is to add viscous terms to the MCAIR explicit Euler method (Reference 7) currently being developed under MCAIR Independent Research and Development (IRAD). The key advantage to this approach is the simplicity of the diagonalized equations, all in wave equation form. This approach has been extended to laminar Navier-Stokes flows and development has progressed to the point of analyzing infinite length channel flows. In this section the basic formulation and advantages of the approach will be described, and preliminary results will be shown.

The basic formulation is composed of Riemann Variables (Q,R) in local streamwise coordinates (s,n) shown in Figure 17. Each equation represents a wave form indicating the proper physical direction in which the four unknowns (Q,R,θ,S) are propagating. The source terms (Z_i) are easily derived and are generally small. The Riemann Equations are derived by combining the streamwise momentum equation and continuity equation (in streamwise coordinates) while making use of the state equation and ideal gas assumption. The normal momentum equation and entropy transport equation are already in wave form.

$$\frac{\partial f_i}{\partial t} + \lambda_i \frac{\partial f_i}{\partial s} = Z_i \qquad \text{— Wave Equation}$$

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{Bmatrix} = \begin{Bmatrix} Q = q + \dfrac{2a}{\gamma - 1} \\ R = q - \dfrac{2a}{\gamma - 1} \\ \theta \\ S \end{Bmatrix} \qquad \begin{Bmatrix} \lambda_1 \\ \lambda_2 \\ \lambda_3 \\ \lambda_4 \end{Bmatrix} = \begin{Bmatrix} q + a \\ q - a \\ q \\ q \end{Bmatrix} \begin{matrix} \text{— Q Riemann Invariant} \\ \text{— R Riemann Invariant} \\ \text{— Normal Momentum} \\ \text{— Entropy} \end{matrix}$$



q = Velocity Magnitude

a = Speed of Sound

S = Entropy

θ = Flow Angle

Figure 17. Riemann Invariant Form of Navier-Stokes Equations in Local Streamwise Coordinates

There are several advantages to this form of solution approach which deserve attention. The equations are minimally coupled since they are in diagonal form. They are only coupled through the source terms of the wave equations and the eigenvalues. The characteristics are unambiguously defined using the local streamline coordinate system. For each equation there is one and only one eigenvalue representing a single wave propagation process. A simple and efficient algorithm gives a time dependent relaxation to steady state for an arbitrary grid. The algorithm is fully vectorizable with vector lengths nearly equal to the number of grid points, making extensive use of CYBER 200 series technology. The formulation has proved versatile (Reference 7) with a host of one, two and three dimensional Euler solutions. Several solutions ranging from incompressible Mach Numbers to high supersonic speeds have been computed. Viscous terms are easily incorporated into the equation source terms.

Development of a laminar Navier-Stokes solver was initiated for the simple case of two parallel plates in motion with constant distance between them. The plates initially move in the same direction at the same speed, then one plate is slowed to rest. The problem is to find a steady state solution and compare to the analytical solution. A two dimensional equation solver was developed first to solve this one dimensional problem, then extended to channel flows.

The parallel plate problem was computed simply with ten computational cells of equal size between the plates. Boundary conditions are composed of the no slip condition and specified plate temperatures. A Reynolds Number of ten based on distance between plates and a Prandtl Number of 0.7 is specified with an upper plate speed at 0.5 Mach Number. Results were obtained in a few hundred iterations which nearly reproduce the analytical steady state solution. Results using extended Riemann Variables for this case were first given in Reference 7. Although the code uses a nonconservative formulation, total mass is conserved to 0.01 percent accuracy for this crude discretization.

A channel flow problem is created simply by perturbing the shape of the lower surface. An 11 by 11 grid discretization is shown in Figure 18, where the lower surface represents one period of a sine wave with a ten percent channel restriction. The flow is periodic with similar boundary conditions as the parallel plate case. The initial flow guess is a linear variation in velocity components from top to bottom whereas entropy and speed of sound are assumed constant. The streamline contours shown are generated after a few hundred iterations representing a simple attached flow solution.

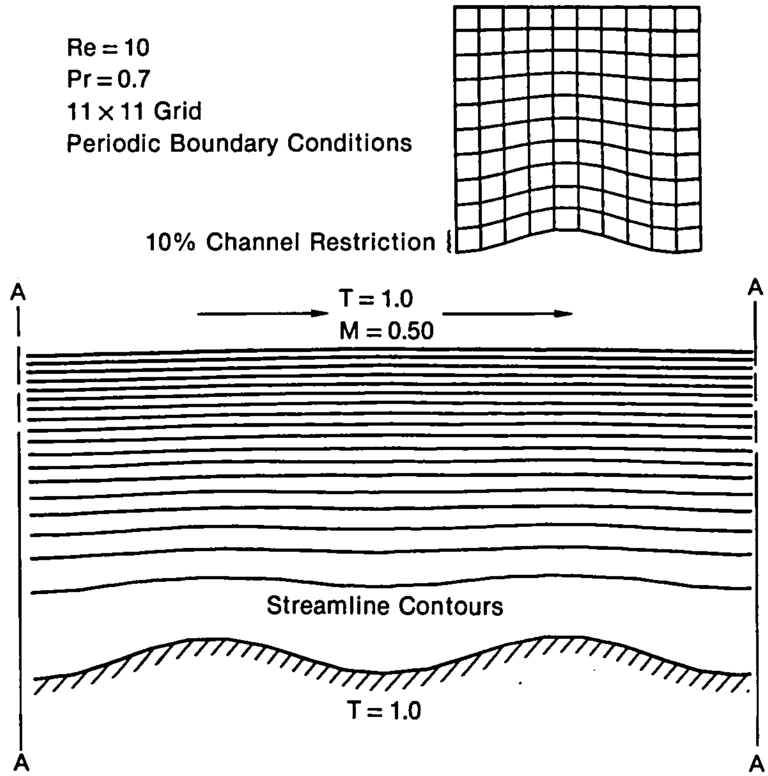


Figure 18. Navier-Stokes Test Problem - Infinite Length Channel With Attached Flow

A separated flow case was calculated by doubling the amplitude of the lower surface sine wave and raising the Reynolds Number to 1000. A refined 41 by 41 grid is used to discretize the flow as shown in Figure 19. The lower surface streamlines are shown at the four time intervals given. The flow was initialized in the same manner as the attached flow case. Each time unit corresponds to the time required for the upper plate to move one half the wavelength of the lower sine wave. After one time unit the flow is clearly separated and reaches its steady state location after two time units. The separation bubble grows in size and strength to that shown after eight time units. This solution represents sixteen thousand constant time steps for each grid point. This developmental code is formally first order accurate in both space and time and requires about one CPU minute per 1000 iterations for this case on a CYBER 203 using full precision operations exclusively.

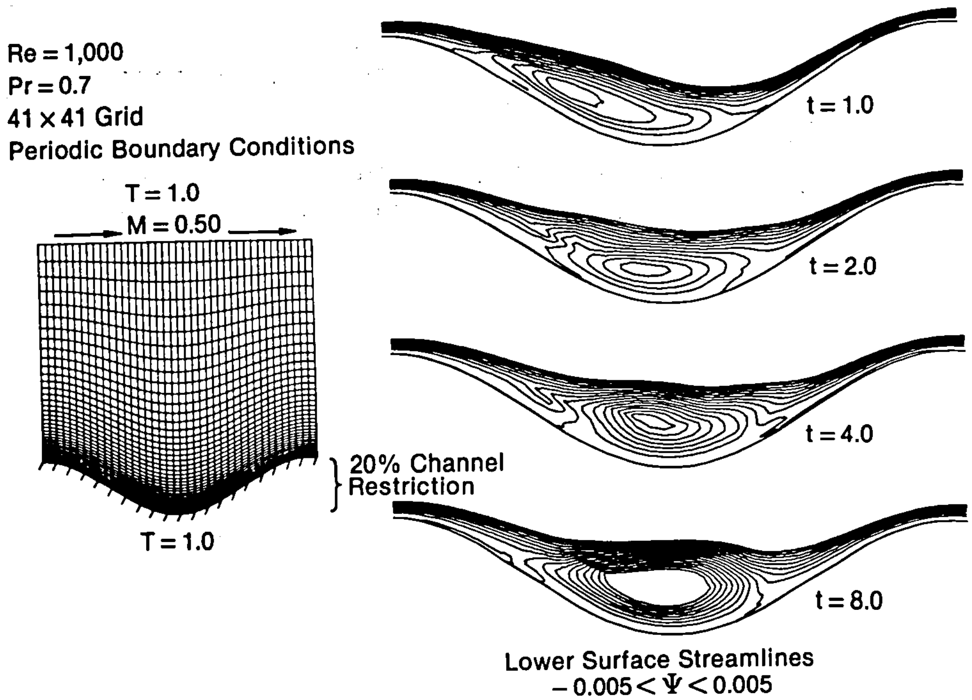


Figure 19. Navier-Stokes Test Problem - Infinite Length Channel With Laminar Flow Separation

The primary goal of this development is to improve the accuracy and reliability over existing methods. The use of Riemann Variables in local streamwise coordinates forms the mathematical system of equations which dictates the proper numerical modeling of the physical wave propagation process. The strong relationship between mathematical, numerical and physical modeling in this formulation is expected to supply the means for an accurate and reliable code. Proper boundary conditions are easily implemented as a result of this formulation.

A secondary goal is to develop a very efficient code. The developmental code has recently improved in efficiency by a factor of about six over the timings given above. The code now has a better streamlined logic and uses half precision (32 bit) operations throughout. Local variable time steps were also implemented which requires fewer iterations to achieve steady state at the expense of time accurate modeling. A further efficiency increase of an order of magnitude may be achieved using higher order methods, multigrid and/or local grid embedding concepts.

Future plans are to develop a Navier-Stokes airfoil code by extending MCAIR's Euler code using information learned from this developmental effort. Extended Riemann Variables will be used for a better shock modeling capability. Both algebraic and two-equation turbulence eddy-viscosity models will be implemented. The point compressible vortex approach will be used for an improved outer boundary condition until a better concept is devised. Subcritical attached flows will be treated initially, followed by flows with leading edge separation. Ultimately this development would be used to analyze wing/body configurations followed by complex aircraft configurations.

SECTION VI
CONCLUSIONS

The panel method couples smoothly with the implicit Coakley code resulting in a significant accuracy increase over the uncoupled code for any given outer boundary location. Three anomalies of the coupled approach are cited as follows: The accuracy of the solution may significantly be effected by changing the relative proximity between panel boundary and outer boundary. Secondly, the outer region is not potential since only some of the boundary values can be specified using potential relations. Finally, an instability results when the panel method is coupled with the explicit MCAIR code. This instability decays when the outer specified boundary values are updated after every few hundred iterations.

The results of this study show significant efficiency improvements can be obtained over the uncoupled approach. Results also indicate the outer potential flow is best represented by the simple point compressible vortex model.

Preliminary development of a Navier-Stokes equation solver using Riemann variables has shown promising results. Infinite length channel flows are computed for attached and separated flow cases with relative ease. The formulation using local streamwise coordinates is simple, accurate and efficient and is recommended to be further developed for airfoil applications.

REFERENCES

1. Ghia, K. N., Osswald, G. A., and Ghia, U., "Analysis of Two-Dimensional Incompressible Flow Past Airfoils Using Unsteady Navier-Stokes Equations," Third Symposium on Numerical and Physical Aspects of Aerodynamic Flows, January 1985.

2. Gilmer, B. R. and Bristow, D. R., "Analysis of Stalled Airfoils by Simultaneous Perturbations to Viscous and Inviscid Equations," AIAA J., Vol. 20, No. 9, September 1982, pp 1160-1166.

3. McGhee, R. J. and Beasley, W. D., "Low Speed Aerodynamic Characteristics of a 13-Percent-Thick Medium-Speed Airfoil Designed for General Aviation Applications," NASA TP-1498, August 1979.

4. Coakley, T. J., "Numerical Method for Gas Dynamics Combining Characteristics and Conservation Concepts", AIAA Paper 81-1257, June 1981.

5. Coakley, T. J., "Implicit Upwind Methods for the Compressible Navier Stokes Equations", NASA TM 84364, May 1983.

6. Coakley, T. J., "Turbulence Modeling Methods for the Compressible Navier Stokes Equations", AIAA Paper 83-1693, July 1983.

7. Verhoff, A. and O'Neil, P. J., "A Natural Formulation for Numerical Solution of the Euler Equations", AIAA Paper 84-0163, January 1984.

8. Thomas, J. L. and Salas, M. D., "Far-Field Boundary Conditions for Transonic Lifting Solutions to the Euler Equations," AIAA Paper 85-0020, January 1985.

9. Morino, Luigi and Kuo, Ching-Ching, "Subsonic Potential Aerodynamics for Complex Configurations: A General Theory," AIAA J., Vol. 12, No. 2, February 1974, pp 191-197.

10. Bristow, D. R., "Development of Panel Methods for Subsonic Analysis and Design," NASA CR 3234, February 1980.

1. Report No. NASA CR-172496	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Study of a Numerical Approach for the Aerodynamic Analysis of Airfoils with Laminar Separation		5. Report Date February 1985
		6. Performing Organization Code
7. Author(s) D. W. Halt, D. R. Bristow		8. Performing Organization Report No.
		10. Work Unit No.
9. Performing Organization Name and Address McDonnell Aircraft Company McDonnell Douglas Corporation P. O. Box 516 St. Louis, MO 63166		11. Contract or Grant No. NAS1-17466
		13. Type of Report and Period Covered Final Contractor Report, Oct 83 - Oct 84
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546		14. Sponsoring Agency Code 505-31-03-03

15. Supplementary Notes

Langley Research Center Technical Monitor - Richard J. Margason

16. Abstract

The objective of this research is to formulate and evaluate a new numerical method for simultaneously and efficiently coupling an external subsonic potential flow and an interior viscous flow such that the two flows match at an interfacing boundary. Both a panel method and a simple point compressible vortex model are used for the outer potential field. The interior flow solvers which were used are the Navier-Stokes and Euler codes of T. J. Coakley (NASA Ames) and the Euler code of A. Verhoff (MCAIR). In order to test compatibility, the panel method is coupled to the less expensive Euler codes since the coupling procedure is identical with the Navier-Stokes code.

The results of this study show significant efficiency improvements can be obtained over the uncoupled approach. Results also indicate the outer potential flow is best represented by the simple point compressible vortex model. The panel method couples smoothly to Coakley's implicit code but is numerically incompatible as coupled with MCAIR's explicit Euler code.

An improved Navier-Stokes code is under initial development which extends MCAIR's Euler code to include the necessary viscous terms. Results are shown for an infinite length channel with one wavy periodic wall with and without laminar separation. Further development is recommended for airfoil applications with the point compressible vortex used for the outer potential flow region.

17. Key Words (Suggested by Author(s)) Computational Fluid Dynamics Navier Stokes Equation Viscous Flow Separation Panel Method (Fluid Dynamics) Euler Equations	18. Distribution Statement Unclassified -- Unlimited subject category 02

19. Security Classif (of this report)	20. Security Classif (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	32	